

Available online at [www.sciencedirect.com](http://www.sciencedirect.com)**ScienceDirect**

Procedia Engineering 81 (2014) 927 – 932

**Procedia  
Engineering**[www.elsevier.com/locate/procedia](http://www.elsevier.com/locate/procedia)

11th International Conference on Technology of Plasticity, ICTP 2014, 19-24 October 2014, Nagoya  
Congress Center, Nagoya, Japan

# Improvement in prediction accuracy by finite element methods of stretch-formed aluminum alloy sheets with a large aspect ratio

Sosuke Sasaki\*, Akira Kono, Susumu Takahashi

*Department of Mechanical Engineering, Nihon University of Industrial Technology, 1-2-1,  
Izumi-cho, Narashino-shi, Chiba 275-8575, Japan*

---

## Abstract

Many parts of an aircraft, such as aircraft fuselage skins, Aluminum alloy sheets are applied for producing these parts. In order to develop dies for stretch forming in a short time, it is necessary to develop forming simulation technologies for prediction. In addition, the skin is deformed by its own weight at all times because of the large but thin shapes of the skin panels. To predict accurately the parts shapes after forming by finite element method (FEM) analysis, not only springback but also own-weight deformation must be considered. The purpose of this study is to improve the accuracy of stretch forming analysis by FEM for sheets with a large aspect ratio, such as those used for airplane skin. For investigating the optimal simulation parameters for predicting springback and own-weight deformation accurately, own-weight deformation test and stretch forming tests with large curvature (1500mm in radius) were conducted. The aluminum alloys A2024-T3 and A7075-T6 are applied for the forming tests. In order to investigate the accuracy of the own-weight deformation and the forming simulation, mesh size was investigated. For own-weight deformation analysis, the simulation results with the mesh size of smaller than 25mm were in good agreement with the experimental results. Regarding springback deformation after stretch forming, the simulation results correspond with experimental results qualitatively for all mesh size.

© 2014 Published by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license (<http://creativecommons.org/licenses/by-nc-nd/3.0/>).

Selection and peer-review under responsibility of the Department of Materials Science and Engineering, Nagoya University

**Keywords:** Springback ; Gravitational deformation ; Stretch forming ; FEM ; Aluminum alloys

---

\*Corresponding author. Tel.: +81-47-474-2322; fax: +81-47-474-2349.

E-mail address: [ciso13024@g.nihon-u.ac.jp](mailto:ciso13024@g.nihon-u.ac.jp)

## 1. Introduction

Recently, the accuracy of aircraft part manufacturing has increased as a result of introducing computer aided design and computer aided manufacturing systems. There have been attempts to manufacture parts using the hole-to-hole assembly method, in which parts are positioned by simply matching index holes [1-3]. Several aircraft manufacturers have succeeded in lowering the costs of assembly jigs by applying this assembly method. However, to use this method, it is necessary to improve the accuracy of aircraft part dimensions, which many aircraft manufacturers have attempted by introducing simulation technologies [4, 5]. Aircraft structures are primarily made of aluminum alloys such as A2024-T3 and A7075-T6, which have a greater springback tendency than steel. Therefore, it is necessary to develop simulation technologies to predict springback.

In particular, it is necessary to develop simulation technologies for stretch forming of aircraft skin for the purpose of designing dies. Because the accurate prediction of aircraft skin shapes and dimensions is a very important parameter in the aircraft assembly process. The occurrence of the springback of aircraft skin is inevitable after forming. Because stretched sheets deform elastically when the two gripping jaws located on both ends are released. In addition, the skin is deformed by its own weight at all times, because the skin panels are large but thin. To predict accurately the shapes of parts after forming by finite element method (FEM) analysis, not only springback, but also own-weight deformation must be considered.

The purpose of this study is to improve the accuracy of FEM analysis of forming for parts with large aspect ratios, such as airplane skin. For investigating the optimal simulation parameters considering springback and own-weight deformation, basic stretch forming tests are conducted. The FEM model results are compared against experimental forming data to determine accuracy.

## 2. Experiment and analysis

First, the accuracy of simulating gravitational deformation is evaluated under conditions of large aspect ratio, which is a characteristic of aircraft parts. Next, springback is investigated by conducting basic stretch forming tests.

### 2.1. Gravitational deformation experiment

An experimental apparatus was set up to conduct gravitational experiments. Fig. 1 shows a schematic illustration of the gravitational deformation experimental apparatus. Tests were performed by using sheets of A2024-T3 condition with a thickness of 1 mm and A7075-T6 condition with a thickness of 1.3 mm. Table 1 lists the material properties of the specimens. Both specimens were square sheets of side length 500 mm. One corner of a specimen of a length of 100 mm and a width of 50 mm was clamped by a jig. Deformation under the specimen's own weight was measured by a laser displacement sensor.

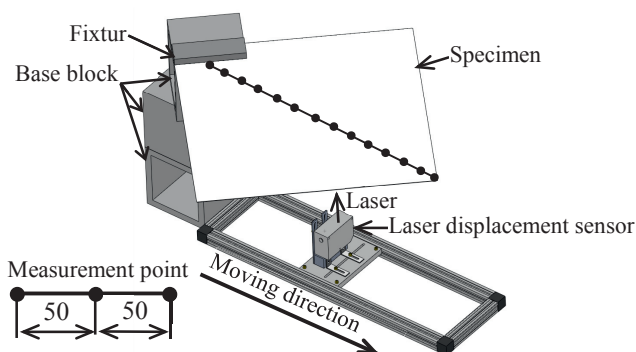


Fig. 1. Schematic illustration of experimental apparatus.

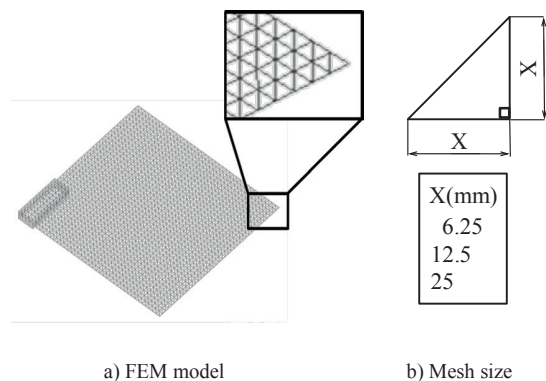


Fig. 2. FEM model and mesh size for sheets.

Table 1. Material properties of specimens.

Materials		A2024-T3	A7075-T6
Young modulus (GPa)		69.9	70.5
Poisson ratio		0.33	0.33
Yield stress (MPa)		325	490
r-value	0 ° (Rolling direction)	0.65	0.55
	45°	0.88	1.06
	90°	0.69	0.54

## 2.2. Gravitational deformation FE analysis

The FEM simulation was performed using a commercial sheet metal forming FEM package, Stampack ver. 6.2.5 (Quantech Corporation). The static implicit method was applied for gravitational deformation analysis. The influences of various mesh sizes on the accuracy of the FEM analysis were investigated by comparing the simulation and experimental results. Fig. 2 shows FEM model and mesh size for sheets. The elements were triangular-shaped shells. The narrow side of the triangular elements was varied from 6.25 to 25 mm.

## 2.3. Basic stretch forming experiment

For investigating the optimal simulation parameters considering springback deformation, basic stretch forming tests were conducted. Tests were performed using sheets of A2024-T3 condition with a thickness of 1 mm and A7075-T6 condition with a thickness of 1.3 mm. Both specimens were rectangular sheets of length 500 mm and width 25 mm. The experimental apparatus and die are shown in Fig. 3. To imitate the state in which both ends are grabbed as in the actual forming process, one end of the specimen is gripped by the chuck of the tensile test machine (AG-100kNX SHIMADZ Corporation) while the other end is fixed on the die with a fixture. The radius of curvature of the specimen after springback and its own-weight deformation were evaluated using the die with a radius of curvature of 1500 mm in the tensile direction.

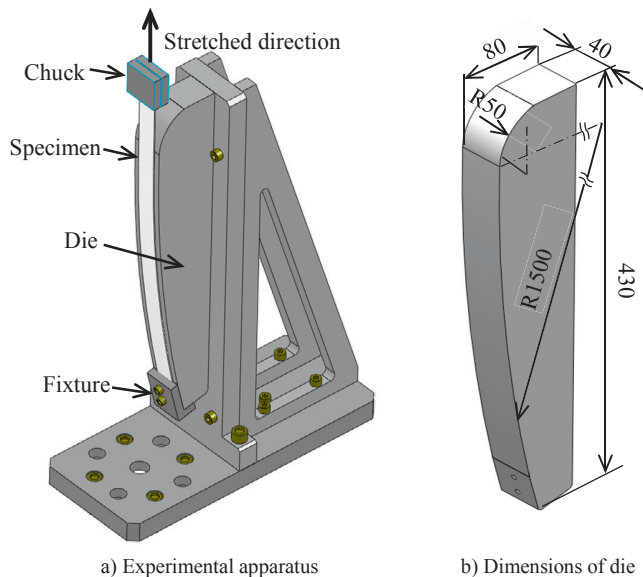


Fig. 3. Schematic illustrations of stretch forming experimental apparatus and forming die.

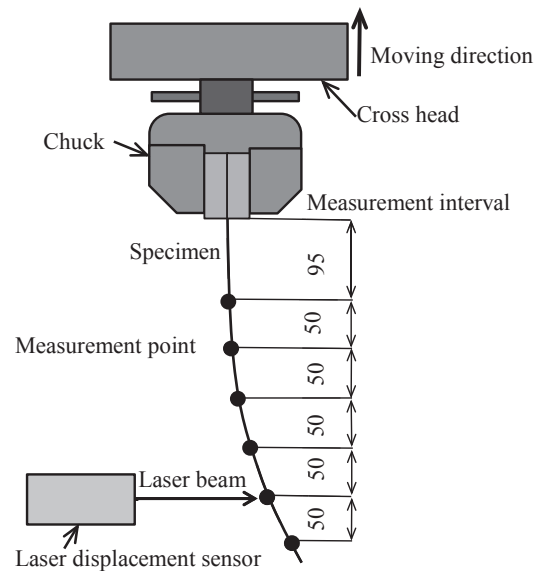


Fig. 4. Shape measurement method.

The initial strain rate was  $1.5 \times 10^{-4} \text{ s}^{-1}$  and elongation was approximately 1.9%. These conditions are the same as those during actual stretch forming of aircraft skin. Curvature was measured after forming. After forming, specimens were fixed to the chucks of the universal tensile test machine. Shape measurement was conducted by laser displacement sensor (LK-H155 KEYENCE Corporation). Fig. 4 shows the shape measurement method. The crosshead of the universal tensile test machine was moved upward. The chuck grip end of the specimen was defined as the reference position, the cross head of the tensile test machine was moved upward, and the horizontal distance between the laser displacement meter and the specimen was measured at various positions of the specimens. Horizontal distance measurement was initiated at a point 95 mm lower than the chuck grip end and conducted at 50mm intervals in the vertical direction. The curvature of specimens was acquired by calculating with three adjacent points (the distances from the chuck grip end were 95, 145 and 195 mm; 145, 195 and 245 mm; 195, 245 and 295 mm; 245, 295 and 345 mm).

#### 2.4. Basic stretch forming FEM analysis

The basic stretch forming FEM simulation was performed by using a commercial sheet metal forming FEM package, Stampack ver. 6.2.5 (Quantech Corporation). Gravitational deformation was considered in basic stretch forming FEM analysis. Fig. 5 shows FEM model and mesh size for sheets. The FEM results were compared against experimental results to determine the former's accuracy. To investigate the effectiveness of the mesh aspect ratio on the accuracy of simulating stretch forming, the mesh size was varied from 6.25 to 25 mm. The dynamic explicit method was applied at the stage of stretch forming and the springback stage was calculated by the static implicit method.

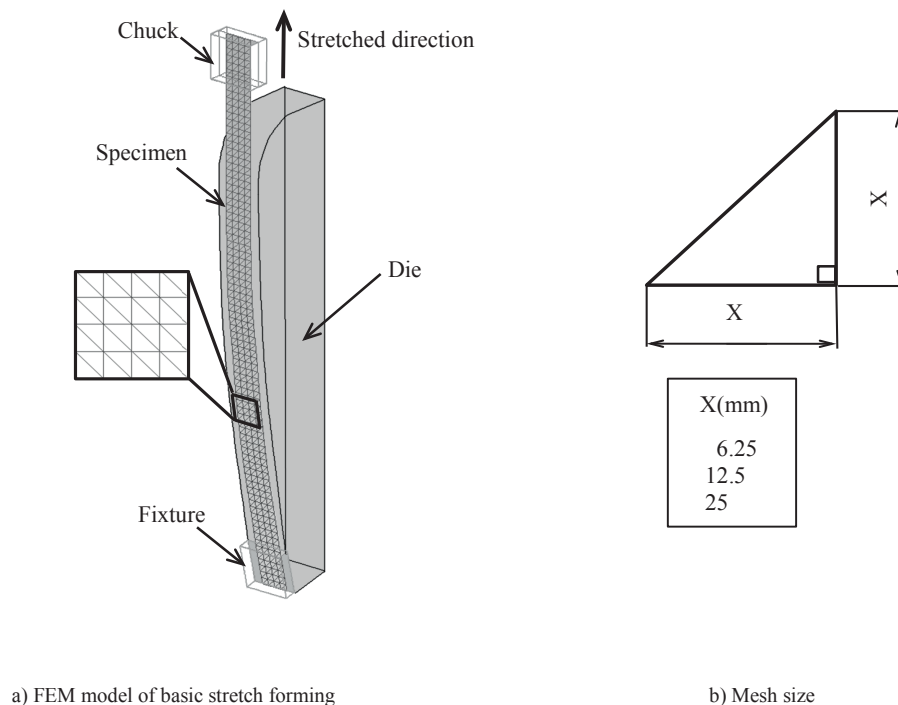


Fig. 5. FEM model and mesh size for sheet.

### 3. Results and discussion

#### 3.1. Gravitational deformation

Fig. 6 shows a comparison of gravitational deformation between experimental and simulation results. The deformation shapes in the simulation, with mesh sizes from 6.25 to 25 mm, agree with the experiment. As the mesh size decreased, the amounts of deformation tended to increase slightly. The influence of aluminum alloy type and sheet thickness on the accuracy of FEM analysis of gravitational deformation was not recognized.

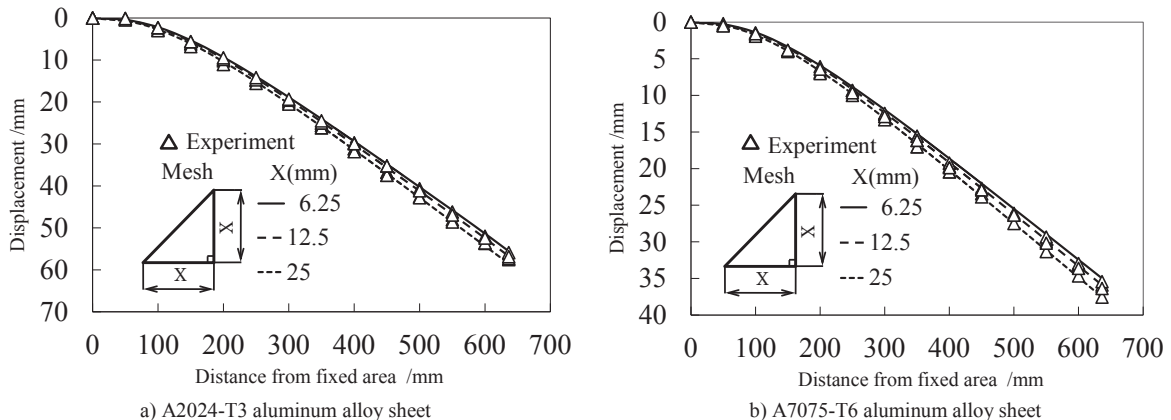


Fig. 6. Comparison of gravitational deformation between experimental and simulation results.

#### 3.2. Basic stretch forming

The radius of curvature the stretch forming FEM analysis at various mesh sizes is shown in Fig. 7. The variations of curvature radius in the basic stretch test FEM analysis, conducted with mesh size of 25 mm, ranged from 80 to 200 mm. On the other hand, the variations of curvature with mesh size of 12.5 mm were the same as those for the mesh size of 6.25 mm. The amount of curvature with both mesh sizes variations ranged from 60 to 130 mm. The tendency for the curvature to vary with decreasing mesh size was not recognized. Also not recognized were the influence of aluminum alloy type and sheet thickness on the accuracy of FEM analysis of stretch forming.

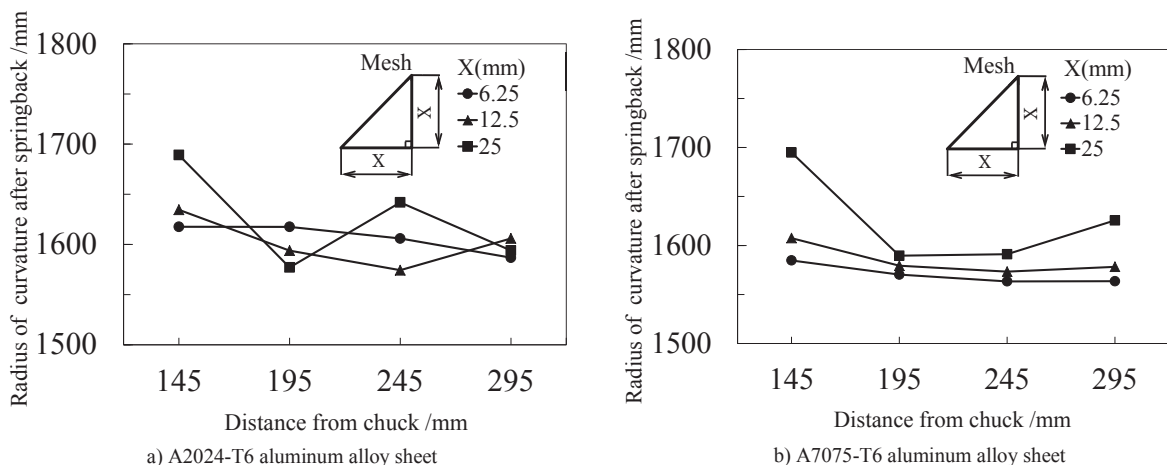


Fig. 7. Radius of curvature of stretch forming FEM analysis at various mesh size.

Fig. 8 shows the curvature radius of basic stretch forming experiment and simulation results for a mesh size of 6.25 mm after springback. The curvature radius of the basic stretch forming test of the A2024-T3 sheet after springback ranges from approximately 200 to 250 mm larger than that of the die. On the other hand, the curvature radius of the A7075-T6 sheet is approximately 200 mm larger. The simulation results of the curvature radius of A2024-T3 are also larger than those of the A7075-T6. Although the simulation result matches the actual results from a qualitative standpoint, it is difficult to quantitatively calculate the actual shape in simulations.

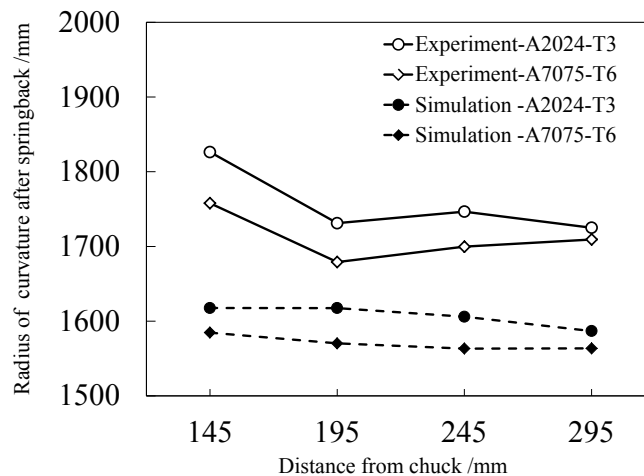


Fig. 8. Radius of curvature of basic stretch forming experiment and simulation with 6.25mm mesh after spring back.

#### 4. Conclusions

In this study, the accuracy of FEM analysis of stretch forming under conditions of a large aspect ratio of specimen shape is evaluated.

Regarding gravitational deformation, simulation results are in good agreement with experimental results for mesh sizes smaller than 25 mm. The influence of aluminum alloy type and sheet thickness on the accuracy of FEM analysis of gravitational deformation was not recognized.

Regarding springback deformation after stretch forming, the simulation results correspond with actual results from a qualitative viewpoint. The influence of aluminum alloy type and sheet thickness on the accuracy of FEM analysis of springback deformation was not recognized.

#### References

- [1] Daniel E. Whitney, MECHANICAL ASSEMBLIES, OXFORD UNIVERSITY PRESS, 2004. New York, 213.
- [2] Muske, S., Application of Dimensional Management on 747 Fuselage, SAE Paper 975605, 1748-1753.
- [3] Hartmann, J., Meeker, C., Weller, M., Izzard, N. et al., Determinate Assembly of Tooling Allows Concurrent Design of Airbus Wings and Major Assembly Fixtures, SAE Technical Paper 2004-01-2832.
- [4] D. Wade, 2001. MSC Software Virtual Manufacturing, 5–6.
- [5] A. Kono, T. Yamada and S. Takahashi, 2014. The effectiveness of FE model for increasing accuracy in stretch forming simulation of aircraft skin panels, The 9th International Conference and Workshop on Numerical Simulation of 3D Sheet Metal Forming Processes, 736–739.